AIR FORCE

QUALIFICATION TRAINING PACKAGE (AFQTP)

for

ENGINEERING

(3E5X1)

MODULE 15

COMPUTER AIDED DESIGN/DRAFTING (CADD)/ GEOGRAPHIC INFORMATION SYSTEM (GIS)

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GEOGRAPHIC INFORMATION SYSTEM (GIS)

Career Field Education and Training Plan (CFETP) references from 1 Apr 97 version.

OPR: HQ AFCESA/CEOT
(SMSgt Randall K. Skinner)

Certified by: HQ AFCESA/CEO
(Colonel Lance C. Brendel)

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INTRODUCTION

Before starting this AFQTP, refer to and read the "Trainee/Trainer Guide" located on the AFCESA Web site **http://www.afcesa.af.mil/.**

AFQTPs are mandatory and must be completed to fulfill task knowledge requirements on core and diamond tasks for upgrade training. ***It is important for the trainer and trainee to understand*** that an AFQTP ***does not*** replace hands-on training, nor will completion of an AFQTP meet the requirement for core task certification. AFQTPs will be used in conjunction with applicable technical references and hands-on training.

AFQTPs and Certification and Testing (CerTest) must be used as minimum upgrade requirements for Diamond tasks.

MANDATORY minimum upgrade requirements:

Core task:
AFQTP completion
Hands-on certification

Diamond task:
AFQTP completion
CerTest completion (80% minimum to pass)

Note: *Trainees will receive hands-on certification training for Diamond Tasks when equipment becomes available either at home station or at a TDY location.*

Put this package to use. Subject matter experts under the direction and guidance of HQ AFCESA/CEOT revised this AFQTP. If you have any recommendations for improving this document, please contact the Career Field Manager at the address below.

HQ AFCESA/CEOT
139 Barnes Dr Suite 1
Tyndall AFB FL 32403-5319
DSN: 523-6322, Comm: (850) 283-6322
Fax: DSN 523-6488
E-mail: ceott.helpdesk@tyndall.af.mil

OPERATE CADD SYSTEMS

MODULE 15 AFQTP UNIT 1

PERFORM CADD FUNDAMENTALS (15.1.1.)

PERFORM CADD FUNDAMENTALS

Task Training Guide

STS Reference Number/Title:	15.1.1. Perform CADD fundamentals
Training References:	• Manufacturers Operators Manual • AutoCAD, reference manual, Microstation reference manual
Prerequisites:	• Possess as a minimum a 3E5X1 AFSC • Completed CDC 3E551B Engineering Journeyman, volume 1, Drafting
Equipment/Tools Required:	• Desktop Personal Computer • CADD Software • Plotter/Printer
Learning Objective:	• Be able to perform basic commands and procedures in the CADD environment
Samples of Behavior:	• Perform basic CADD functions
Notes:	
• This is a guide to basic CADD fundamentals and concepts. It is recommend trainees complete the appropriate manufacture's software tutorial to fully comprehend CADD operations. This guide is not built on any one CADD platform nor will it fully train anyone on a specific CADD engine.	

PERFORM CADD FUNDAMENTALS

Background: Computer Aided Design and Drafting (CADD) is a powerful tool increasing the designer and drafters ability to develop complex construction designs. This tool can only reach its full potential if the operator has a good working knowledge of basic drafting fundamentals. Consider CADD an electronic drafting board allowing objects to be created at a one to one scale. This provides the user the capability to plot or view drawings at any scale. Additionally CADD software allows for quicker and easier manipulation of production drawings. Over the years CADD has evolved from a DOS-based, command-line driven program to a full-fledged Windows application. With the move to a Windows environment the CADD software platforms are becoming more and more familiar. Thus once basic CADD fundamentals are mastered the switch between different software is seamless. Although some of the references are representatives of the AutoCAD software it is the intention of this QTP is to be software generic. Today, CADD is prevalent in all areas of the engineering world from the office environment to critical contingency operations. In reality, the CADD programs of today have improved the talents and characteristics of drafters, designers, and machinists, not replaced them. This QTP is broken down into two sections. The first section will provide a general overview of CADD fundamentals, while the second portion provides CADD fundamentals from an AutoCAD standpoint. However personnel using other platforms should not ignore this section as the majority of the commands explained pertain to all CADD software manufactures. Moreover the performance exercise at the end of the QTP can be accomplished using any CADD platform. A third section covering Microstation will be included in a future version of this QTP.

NOTE: The trainee should be knowledgeable of the Windows environment before proceeding.

SECTION I: Generic CADD Fundamentals and Overview

Step 1: CADD Drawing Editor Familiarization

One of the first things accomplished is to become familiar with the CADD screen and how to communicate within the CADD environment.

- CADD program window is divided into five parts:
  - Pull-down menu bar
    - Typically located along the top of the screen
    - Offers pull down menus with a variety of commands
      - ✓ Selected in typical widows fashion
  - Docked and floating toolbars
    - Menus are considered docked when in the default position
    - Menus can be moved around in drawing editor – this is floating
  - Drawing area
    - The work space
    - Occupies most of the area
  - Command window
    - Located at the bottom in AutoCAD
    - Located at the top of the screen in Microstation
    - Input typed commands
      - ✓ "ESC" is an important command to remember, it breaks current command
  - Status Bar

Step 2: Specifying Point

As the mouse is moved around the screen cross hairs appear. This is the drawing cursor. There are several different ways to specify points the CADD environment.

- Picking a point on the screen with your pointing device.
  - Move cursor around screen
  - Place cursor in the middle of the screen
  - Press and release left mouse button
    - ✓ Picked first point
- Typing in absolute coordinates on the keyboard.
- Typing in relative coordinates on the keyboard.
- Picking a point on the screen using the keyboard's arrow keys.
- Using object snap to specify a point based on existing geometry.

Step 3: Pull-Down Menus

Like most Windows programs, the pull-down menus available on the menu bar offer easy-to-understand way to access the general controls and settings. Within these menus you'll find the commands and functions that are the heart and soul of the CADD platform.

- Pull-down menu options perform four basic functions:
  - Display additional menu choices
  - Display a dialog box that contains settings you can change
  - Issue a command requiring keyboard or drawing input
  - Offer expanded set of tools

Step 4: Toolbars

Pull-down menus offer a full range of easy-to-understand options. However they require some effort to navigate. Toolbars, on the other hand, offer quick, single click access to the most commonly used toolbars. A tool tip appears just below the arrow cursor when cursor rests on the tool to help the operator understand each tool.

- Toolbars perform three types of action:
  - Display further options
  - Open dialog boxes
  - Issue commands requiring keyboard or cursor inputs
- Flyouts
  - Most toolbar tools start a command as soon as you click them, but other tools display a set of additional related tools – these are called flyouts
- Mobility
  - Toolbars can be docked on the top, bottom or sides of the screen
  - Toolbars can be positioned free floating on the drawing editor screen

Step 5: Starting a CADD Drawing

The CADD drawing is opened a couple of different ways. Most CADD engines have a startup dialog box that allows the operator to select a new or existing file. Additionally some CADD platforms have start-up wizards that automatically step-up drawing with standard user functions.

Step 6: Viewing Options

Once in the drawing editor of the CADD platform operator can easily manipulate the viewing capabilities by using the zoom function. Several options are available under the zoom functions. This function can be accessed several different way in all CADD platforms.

Step 7: Drawing and Modifying

Use the drawing toolbar to create an object while in the drawing editor screen. All CADD software basically has the same drawing tools loaded in the toolbar menus. Additionally all CADD platforms have a modify toolbar which is used to manipulate the objects.

- **Drawing Toolbar**

  Here is a list of some of the drawing aids normally associated with the drawing toolbar:
  - Line
  - Multiline
  - Rectangle
  - Arc
  - Circle
  - Eclipse

  This is by no means an all-inclusive list.

- **Modify Toolbar**

  Here is a list of some of the modifying aids normally associated with the modify toolbar:
  - Erase
  - Copy
  - Mirror
  - Offset
  - Rotate
  - Scale
  - Trim
  - Scale

  Again, by no means is this an all-inclusive list.

Step 8: Text

One of the more tedious drafting tasks is applying notes to your drawing. CADD allows notes to be easily added in a standard professional manner. CADD offers a wide range of text formatting options, such as controlling fonts, text height, justification, line spacing, and width. Most CADD platforms now even include a spell checking function.

Step 9: Dimensions

Dimensioning is vital to how well a design works and how quick it develops. With CADD, the operator can easily add tentative and final dimensions to any drawing. CADD provides an accurate dimension without having to take measurements. A user simply picks two points to be dimension, the dimension line location, and the CADD software will do the rest. The software can automatically update dimensions whenever the size or shape of the dimensioned objects changes. The CADD dimensioning feature has many settings giving the operator an enormous amount of flexibility.

Step 10: Plotting

Plotting or printing is handled in the CADD environments by a dialog box where the appropriate setting are indicated. This dialog box allows the user to specify the paper size, scale of the plot, type of printing device and what portion of the drawing is to be printed.

SECTION II: AutoCAD Fundamentals and Overview

Preface: There are some particular features about the AutoCAD Drawing Editor that should be understood before attempting command entry. First, a command cannot be entered unless the program ready indicator reads, "Command." This prompt will only be present if no other command or routine is in progress. Only valid AutoCAD commands and defined functions will be recognized at this prompt. If the command prompt is not present, then the program is processing an operation or routine. You can cancel a command by pressing the "ESC" key on the keyboard. However some commands requires this more than once. Certain commands have previously assigned values maintained for future use. These values are known as "defaults." When a command has defaults, it's displayed within angle brackets < >. To use these defaults just respond with "Enter" or "Return." Some commands will provide a group of "options" contained on the prompt line. Options will have several capital letters in the word. You need only to type that capital letter in response.

The Space Bar performs the same function as the "Enter," "Return," or the right button on the "mouse", but will function as in a typewriter when typing TEXT.

Commands that tend to be repeated several times can be called with the word "multiple" preceding the actual command name (i.e., "Command: MULTIPLELINE" [MLINE] for successive LINE commands). Using the "ESC" key can stop this function. Ensure the full menu is displayed by typing MENU in the command line and pick ACADMENUFULL. Detail instructions can be found in HELP or program Tutorial, which is an excellent learning tool in the training environment.

Coordinate systems There are two basic types of coordinate systems used in AutoCAD, *world* and *user*.

The *World Coordinate Systems* (WCS) is a fixed coordinate system and is common to all drawings created in AutoCAD.

The *User Coordinate System* (UCS) is an arbitrary coordinate system defined within the WCS. It has an arbitrarily assigned orientation and origin.

Specifying Point There are several different ways to specify points in AutoCAD including:

- Typing in absolute coordinates on the keyboard
- Typing in relative coordinates on the keyboard
- Picking a point on the screen with your pointing device
- Picking a point on the screen using the keyboard's arrow keys
- Using object snap to specify a point based on existing geometry
- Employing a combination of these techniques using X/Y/Z point filters to compose one point from several specifications

When AutoCAD asks for a point, it expects you to enter the coordinates of a location in the drawing. The format of the prompt for points is ***Prompt: point***. Points are the most common types of data you'll supply to AutoCAD. Ordinarily, points are expressed in relation to the current UCS as set by the UCS command.

Many AutoCAD drawing and editing commands accept three-dimensional points (*x,y,z*). However, you can often omit the Z value, and AutoCAD fills in the value you've established as the current elevation.

After you specify the point, a small marker appears at the location for your reference. Such markers, or blips will disappear when you regenerate or redraw the display.

Note: If LIMITS checking is enabled, AutoCAD checks all points you enter to see if they lie outside the drawing limits. If you try to draw outside the drawing limits, AutoCAD displays the **Outside limits** message and rejects the point. The LIMITS command, controls the limits check and the drawing boundaries themselves.

Absolute Coordinates You can enter absolute coordinates using any of the following formats:

Cartesian You can specify a point by typing its X, Y, and Z values (in decimal, fractional, or scientific notation) separated by commas. For example, to specify the point with an X coordinate 3.5, a Y coordinate of 7.25, and Z coordinate of 4.75, you'd enter: **3.5,7.25,4.75**. To specify the same point using fractional notation, you'd enter: **3-1/2,7-1/4,4-3/4**. If you omit the Z value, the current elevation is used as the Z value. Thus you could enter: **3.5, 7.25** to designate a point with the same X and Y coordinates as in the previous example, but with a Z coordinate equal to the current elevation.

Polar You can specify a point by entering its distance from the current UCS origin and its angle in the XY plane, separated by <. For example, to specify a point at a distance of 8.5 drawing units from the UCS origin at an angle of 45 degrees relative to the UCS X axis (in the UCS XY plane), you'd enter this: **8.5<45**

Cylindrical The cylindrical point format is another 3D variant of the polar format. It describes a point by its distance from the UCS origin, its angle in the XY plane, and its Z value. The distance and angle are separated by <, and the angle and Z value are separated by a comma. For example, to specify a point at a distance of 7 drawing units from the UCS origin, at an angle of 45 degrees relative to the UCS X axis (in the UCS XY plane), and with a Z coordinate of 5.5, you'd enter this: **7<45,5.5**

Relative Coordinates As noted earlier, absolute coordinates are relative to the current UCS origin. To specify a point as an offset from the last point you entered, simply type an @ before the rest of the specification. For instance, if the last point specified was (10,6,4), then entering: @2.5,-1.3,1.5 is equivalent to specifying the absolute coordinates (12.5,4.7,5.5). In relative coordinates, an unspecified Z value is taken to be zero. Therefore, if the last point you entered was (10,6,2), the relative coordinates: @2.5, -1.3 would be equivalent to specifying the absolute coordinates (12.5,4.7,2).

You can use any of the absolute coordinates formats (Cartesian, polar, spherical, or cylindrical) in a relative specification. For example, if the last point you specified was (10,6,2), entering either: @0.3 or @3<90 would be equal to the absolute coordinates (10,9,2).

Last Coordinates Entering @ by itself is shorthand for the relative specification "@0,0,0" or "@0<any-angle." It specifies a zero offset from the last point. For instance, if the last point you entered was (5,5,5), entering just @ specifies point (5,5,5) again. AutoCAD stores the 3D coordinates (x, y, z) of the last point entered. Whether the Z value is used when you choose this method to enter a point depends on whether the command accepts a Z coordinate value.

Drawing Setup *LAYER* - The AutoCAD program has a feature allowing the objects of the drawing (entities) to be placed on separate layers if needed. There are no limitations to the amount of layers that are in one file, though there cannot be two layers with the same name. You can "turn on and off" these layers at will. So, if you are creating house plans, you can have each plan (plot, foundation, floor, electrical, plumbing and mechanical) on separate layers that can be overlaid. This helps eliminate conflicting information between different design functions and ultimately reduce costly field contract modifications. Additionally objects can be further separate by identifying line types and line colors by layers.

Setting UNITS of measurement This command controls the type of units of measurement and precision of their display for each drawing. These units include decimal, scientific, architectural, and engineering notations. This command will also determines the particular format of angles in the drawing, including the dimensions and direction, angles will turn (clockwise or counter clockwise) and starting point (i.e., 3 o'clock = zero degree or 12 o'clock = 90 degrees).

Drawing Aids The AutoCAD program has a few commands assisting the operator with the object creation. These drawing aids can be used at any time during the drawing process. The basic drawing aids are the SNAP, GRID, and ORTHO commands.

- **Snap** This command will control the movement of the crosshairs to the smallest drawing increment expected. The operator can specify points on the drawing at a preset "snap" SPACING when using a pointing device (a mouse). Entered value can be any number that compiles with the current "units" setting. SNAP can be turned ON or OFF any time by using the F9 function key.
- **Grid** This command allows you to set the "grid dots" spacing. By default AutoCAD sets the grid spacing to match the SNAP setting, but you can override this setting. This comes in handy in precision drawings with predetermined increments.
- **Ortho** This commands turns "on" the Ortho mode, which will help you position objects at 90-degree angles in relation to the crosshairs.

NOTE:
These initial settings can be saved as a Prototype drawing for future projects

Text The TEXT and DTEXT command can be used to place text (words) and "notes" on the drawing. When typing a sentence, the entire sentence is considered to be one entity that can be moved and sized. Text can be justified to be ALIGNed between two points you are prompted to specify, FIT, CENTERed or right justified. You can change the text style with the DDSTYLE command. With pointing device, click point to where you want your text to begin and type. Text can be edited by using the DDEDIT command brings up text in a dialogue box.

HINT:
You can type text in Microsoft Word, copy text to the clipboard, open MTEXT dialogue box in AutoCAD drawing (CTRL-V) and paste

In-Line Text Drafting Symbols When you want to include commonly used drafting symbols with your text that are not on the computer keyboard, as in the plus/minus sign or degree symbol, or underline a string of text, there are control codes you can use. By typing these symbols IN the string of text, AutoCAD will automatically "convert" them to the respective symbol. Each of these control codes is described as follows, but must be preceded by two percent signs to activate them, as shown:

- %%p-This will result in the plus/minus sign
- %%d-This will result in the degree symbol
- %%c-This will result in the diameter symbol
- %%u-This will turn on (and off) the underline feature
- %%o-This will turn on (and off) the overscore feature
- %%n-This will result in the "n" special character to be displayed

Basic Drawing Command There are several ways to access commands including: Icons, Pull-down menus, toolbars or typing the command in the command line. TIP: There are clues in every prompt that can save time of keyboard entry. The capital letters in prompts need only to be keyed in to respond; i.e., <exit> just type "X". (lower or upper case).

- **Erase** Select object to erase, it becomes "highlighted," then ENTER.
- **Undo** Used to undo previous commands.
- **Redo** Used to redo an undo (the last UNDO only).
- **Line** The line command is one the most fundamental drawing entities. TIP: Direct Distance Entry Lines- Click start point "drag" in the direction you want it to go then type the distance (length) you want the line to be and enter. MLINE is handy for drawing walls. To undo a line simply type "u" after the "To point:" prompt. If you are drawing a closed polygon such as a square, you can specify three sides and then at the "To point:" type "C" and the line will automatically close on itself (the fourth side). If you hit ENTER after making a line, you can continue a line from the last segment of the most recent LINE command, you can simply make the LINE command active and hit ENTER key at the "From point:" prompt. The line will automatically attach this new line segment with the last one made. To create 3D objects you can use the PLINE command and MLINE command and RENDER to "fill in" area.

Creating Arcs And Circles

- **Arc** The arc command creates a portion of a circle. You must specify three points of the arc before AutoCAD can draw one. Points to choose from are the Center, starting Direction, End point (of arc), Length of chord, Radius, or included Angle. When the three points of the arc have been inputted, the arc will be drawn and returns to the "Command" prompt. For continuation feature as with the LINE command, simply enter the ARC command again and follow same procedure. It will maintain its tangency to the previous arc unless overridden by using the "Direction" option.

- **Circle** The circle command creates circles using any of five options:
  - **Center and Radius (default)** - Specify the "center point", then the radius. Both of these pieces of data can be input as typed in coordinates, or you can use the pointing device to locate each feature. You can use the pointing device to "drag" the circle image to the desired radius. If you have the SNAP "on" and the Coordinates "on" then you can use the X, Y, values in the "status bar" to guide you while creating this circle.
  - **Center and Diameter** - This allows the operator to create a circle by specifying the Center of the circle, and the diameter. The "dragging" option can also be applied.
  - **Center and Diameter** - This allows the operator to create a circle by specifying the Center of the circle, and the diameter. The "dragging" option can also be applied.
  - **Three Point Circle (3P)** - This allows the operator to create a circle simply by specifying any three points "on" the perimeter (the circumference)
  - **Two Point Circle (2P)** - This is the same as the three point instead the two points are automatically diametrically 180 degrees apart.
  - **Tangent, Tangent, Radius (TTR)** - Allows the operator to create a circle by specifying two objects to which the circle will be "tangent," and specifying the radius.

Dimensioning objects DIM You have several dimensioning styles you can pick from in the Dimensioning Styles dialogue box (DIMSTYLE). The default is Standard, but you can have Geometry, Mechanical, etc., or create your own from the Data menu by picking Text Style and save it for future use. From the Dimensions toolbar select Linear, Radius or Angular Dimensions. With the Snap option "on" this can help. TIP: The DIMFIT command can move leader and text independently.

Controlling Your Display Screen There are a number of commands that allow control the display and position the "camera" in various locations around your drawing. You can magnify, pan, and redraw drawings and the time to do such functions. The basic commands are ZOOM, PAN, REDRAW, and REGEN.

- **Zoom.** The ZOOM command magnifies the drawing based on the option chosen:
  - **All** - This option will cause the entire drawing to be displayed onto the screen based on the LIMITS setting.
  - **Center** - This option allows the operator to specify a point on the existing drawing that will be the center of the display, then prompts for a magnification value.
  - **Dynamic** - Will display a "view box" around object that can be sized and moved to what you want displayed.
  - **Extents** - Similar to the ALL option except it disregards the LIMITS setting.

- **Window** - Allows operator to create a ZOOMing window by specifying two opposite corners.

- **Pan** The PAN command can be used to move or shift the viewing area of the current display screen. It does not reposition the object in the drawings, only the viewing area. Pick a point to "grab" on, and then pick a point to "drag" to.

- **Redraw** This command will "refresh" the screen removing "blips" as well as redrawing objects that were partially hidden.

- **Regen** This command "regenerates" your drawing (excluding the "frozen" Layers) and then redraws the entities in the current display. Sometimes REGEN will automatically happen when you ZOOM or PAN.

- **Viewers** This command allows you to control the accuracy of the display of circles, arcs, and ellipse. This also affects the speed at which they will be displayed. The default value is 100. Higher the value, up to 20,000, the more accurate these items become when ZOOMed in, but less speed in bringing them up. This does not affect the plotting/printing product that will be the most accurate no matter the setting.

Creating a Symbol Library WBLOCK - This block command allows the operator to create a block definition (symbol) and save it as a file to a symbol library subdirectory for future use (insertion) in other drawings. First create your symbol then make the WBLOCK command active, name it, then choose where you want the base point for insertion. To insert block, make the INSERT command active. To redefine a block, activate WBLOCK with same name and it will prompt you if you want to redefine it. Type "Y" for yes.

Attributes Attributes are entities that contain textural information that can be bound to a block and inserted into a drawing. The information can be changed with each insertion of the block making them very powerful and versatile. Some uses include control numbers (an attribute) to computers (a block) in an office floor plan (drawing). First you must define it with the DDATTDEF command. You can edit the dialogue (DDEDIT), the attribute (ATTEDIT) or both (DDATTE).

Advanced Edit Commands

FILLET Given a polyline image, you can place a radius fillet at each "vertex" by selecting this option and providing the radius.

CHAMFER This command can be used in the same manner as the fillet command. Provide chamfer distances.

STRETCH This command allows you to move a selected portion of a drawing, preserving connections to parts of the drawing left in place. Lines, arcs, solids and polylines can be stretched.

ARRAY This command allows you to make multiple copies of selected objects either around a specified pivot point, or in rows and columns arrangement.

MIRROR This command can be used to "flip" a selected group of entities or duplicate them "about" a mirror "fold" line. TIP: If you want to include text, set MIRRTEXT to 0 before executing MIRROR command. This will make the text legible and not "flipped."

HATCH The boundary hatch command allows you to draw selected crosshatch patterns within a boundary area. You have the option to choose a pattern, style, and the angle to be drawn.

TRIM This command can be used to "cut off" any portion of a Line, Arc, or Circle which allows you to change the length of the entity you select. TIP: To save time zooming in to find the correct cutting edge, when prompted to select cutting edges just hit ENTER to select all edges

EXTEND This command can be used for the opposite purpose than trim in that it lengthens entities. Same TIP applies to the selection of edges.

SCALE Used to change the size of any selectable object.

HINT:
One step MOVE or COPY When prompted for base point or displacement, do not ignore the last two words of this prompt. Displacement is simply how much an entity will move in each direction in the X, Y axis (e.g., 2,0 on the command line will move it 2 units to the right).

CAL On line Geometric Calculator used to solve mathematical problems or locate specific points in a drawing while in AutoCAD.

Print/Plot Configuration Dialogue Box Allows operator to predetermine various print/plot settings.

- **Device and Default Selection** - Lists the current "default" printer or plotter. Up to 29 separate devices.
- **Pen Parameters** - Assigns specific plotter pens to drawing colors, changes pen speed, line type, and widths.
- **Additional Parameters** - Specify how much of the drawing (rectangular display) to be plotted.
- **Paper Size and Orientation** - Specify sheet size to be print/plotted on and the units.
- **Scale, Rotation, and Origin** - Allows you to rotate the drawing on your sheet of paper, change the lower left plot origin (0,0), and assign a plot scale for your drawing
- **Plot Preview** - Allows operator to preview the drawing, as it would look on the assigned paper sheet.

Saving Drawings Do not forget this step! Set the auto save feature so that interruptions such as power outages, evacuations, or human error will not ruin your day or time lines.

Using the preceding information and the manuals provided with your CAD software, complete the Double light switch cover drawing below:

Step 1: Start the Windows environment

Step 2: Click on the CAD (AutoCAD 2000) icon

Step 3: Open a new drawing under FILE

- Choose new, the Create New Drawing dialog box appears
- Click the use Wizard button at the top
- Select Quick Setup from the list box
- Click Ok. The Units dialog appears use default settings
- Enter page width 24, length 36
- Click finished. A new drawing appears in the AutoCAD window

Step 4: "LINE COMMAND"

- From the **DRAW** floating toolbar
- Select the **LINE** icon flyout
- Select the **LINE** Icon
- Command prompt sequence:
  A. From point: 2,2 (Absolute Coordinate)
  B. To point: 6.5,2 (Absolute Coordinate)
  C. To point: @0,4.5 (Relative Coordinate)
  D. To point: @-4.5,0 (Relative Coordinate)
  E. To point: Type C (Will close the shape)

When drawing lines, AutoCAD displays a rubber band line showing where the next line segment will go. Tell AutoCAD where to place the endpoint of the next line by moving the crosshairs to a new location on the screen and pressing the pick button on your mouse (LEFT button). The more correct and accurate method is to input coordinates (Absolute, Relative, or Relative Polar). The outline of the switch cover is complete. Next draw the mounting holes and switch cutouts.

Step 5: "CIRCLE COMMAND"

- From the **DRAW** floating toolbar
- Select the **CIRCLE** icon flyout
- Select **CIRCLE, CENTER, RADIUS** icon
- Command prompt sequence:
  A. 3P/2P/TIR/<Center point>: 3.38, 3.13 (Absolute Coordinate)
  B. Diameter/Radius/<Radius>: 0.125
  C. Press ENTER to exit and complete the **CIRCLE** command

At the last prompt, **Diameter/<Radius>:** you have a choice of entering either the radius or the diameter of the circle. Radius was enclosed by angle brackets < > to indicate that it was the default choice. AutoCAD assumed the number you were going to enter would be the radius of the circle. In order to enter a diameter instead, go to the keyboard and enter the letter **D,** then press ENTER; this will switch the default Radius command to Diameter, which will enable you to enter the diameter of a circle instead of the radius.

The first mounting hole for the switch cover is now complete. Next you can make a copy of the circle by using the Copy command. But before you do, we'll introduce two very important features here: Selection-sets and OBJECT SNAPS.

SELECTION-SET Many editing commands in AutoCAD ask to select the objects to be edited. To specify these objects, create a selection-set. A selection-set is one or more objects identified to perform specified editing operations at one time. When the prompt **Select objects** appears, AutoCAD is asking you to select the objects for edit. As many objects as needed, can be selected in selection-set. When finished selecting objects, press ENTER. There are three ways to create a selection-set in AutoCAD. The next demonstration will show how selection-set is used.

OBJECT SNAP An object snap override is used to select a specific geometric point on an object, such as the center point of a circle or the midpoint of a line. This feature is important because it combines two benefits with one pick of the pointing device: speed and a very high degree of accuracy. There are several object snaps to choose from. Holding down the shift key and clicking the RIGHT mouse button will display the Object Snaps that are available on your screen.

Step 6: "COPY COMMAND"

- From the **MODIFY** floating toolbar.
- Select **COPY** icon.
- Command prompt sequence:
    A. Select objects: (Select the object you want to copy)
    B. Select objects: 1 selected, 1 found
    C. Select objects: Press ENTER to accept the selection-set
    D. <Base point or displacement>/Multiple: Select **CENTER** object snap (to specify the center of the circle as the Base point)
    E. Second point of displacement: @2.24<90 (Relative Polar Coordinate), ENTER

The two mounting holes are complete. Now it's time to draw the switch cutout. To do this, draw a rectangle 0.38 unit wide by 1.0 unit long. You can use the same method you used previously to draw the first three sides of the cover. To draw the fourth side use the Close option. Follow the prompts below, but be sure you understand each method of point placement as you enter them. If a mistake is made, enter U from the keyboard at the next prompt to undo it.

Step 7: "LINE COMMAND"

- From the **DRAW** floating toolbar
- Select the **LINE** icon flyout
- Select the **LINE** icon
- Command prompt sequence:
    A. From point: 3.19, 3.75 (Absolute Coordinate)
    B. To point: @0.38<0 (Relative Polar Coordinate)
    C. To point: @1<90 (Relative Polar Coordinate)
    D. To point: @0.38<180 (Relative Polar Coordinate)
    E. To point: Type C (to automatically close the rectangle), ENTER

The switch cover is almost complete. You can create a mirror image of the holes and rectangle using a drawing aid called Ortho mode.

ORTHO MODE Before going any further, select the F8 key on your keyboard or double click on the **ORTHO** block on the status bar at the bottom of the screen. This will turn on Ortho mode, which will help you position objects at 90-degree angles in relation to the crosshairs. You may turn Ortho mode on or off any time you like, even while other commands are in progress. Notice the word Ortho appears in the status line on your display screen.

Step 8: ‘‘MIRROR COMMAND”

- From the **MODIFY** floating toolbar
- Select **COPY** object flyout
- Select the **MIRROR** icon
- Command prompt sequence:
    A. Select objects: Use selection-set method and pick both circles and the rectangle
    B. First point of mirror line: Select **MIDPOINT** object snap and pick either line that makes up the top and bottom edges of the Cover
    C. Second point: Rotate mouse until mirrored objects are to the RIGHT of the originals and pick the LEFT mouse button
    D. Delete old objects? <N> Press **ENTER** to keep the original objects

The light switch cover is complete. All that is left is to redraw the display screen to refresh your drawing. If you wish to save this drawing, from the **FILE** pull-down menu, pick **SAVE** and at the Save dialog box, type COVER. Pick OK to activate this command.

Step 9: “REDRAW COMMAND” This command is used to remove the blip marks from the display screen and refresh the drawing screen.

- Select **VIEW** menu
- Select **REDRAW VIEW** command or,
- Select the **REDRAW** icon from the standard toolbar

NOTE:
If you have completed the task properly, your drawing should look like figure 1.

Step 10: “ERASE COMMAND” To remove objects from the screen, use the Erase command

- From the **MODIFY** floating toolbar
- Select **ERASE** icon
- Command prompt Sequence:
    A. Select objects: Using selection-set method, pick the entire Cover drawing
    B. Select objects: ENTER, to complete the command sequence

Step 11: "<u>UNDO COMMAND</u>" To reverse the effects of the Erase command, or any other command, enter the **UNDO** command. This command allows you to experiment with commands and features in AutoCAD without having to keep the results. The Undo command has several options and can be a useful tool when not sure how a particular command will act. To bring back the cover, use the Undo command to reverse your last AutoCAD command.

- Select **EDIT** menu.
- Select **UNDO** command or
- Select the **UNDO** icon from the standard toolbar.

Step 12: "<u>REDO COMMAND</u>" This the only tool that lets you undo an UNDO command.

- Select **EDIT** menu
- Select **REDO** command or,
- Select the **REDO** icon from the standard toolbar

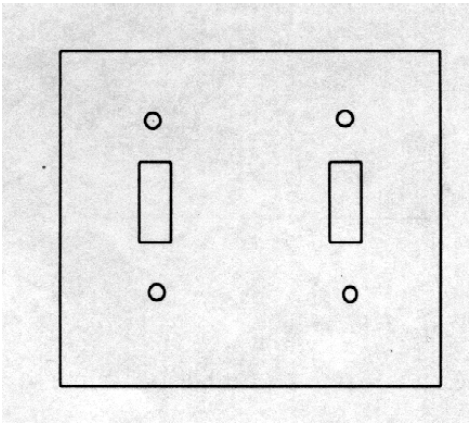


Figure 1:

Step 13: "<u>EXERCISE</u>" Now it's time to practice what you have learned. Draw, to scale, the floor plan of your office that you manually drafted in AFQTP 3E5X1-14.

Review Questions
for
Perform CADD Fundamentals

Question	Answer
1. What are the five parts of the CADD program window?	a. View, Drawing editor, toolbars, menu bar and status bar b. Pull-down menu, toolbars, Drawing area, Command window, Status Bar c. Sill, Screen, header, frame, and shutters d. Pull-down menu, Toolbars, Drawing area, Status menu and Help menu
2. What toolbar contains the circle function?	a. Modify b. Draw c. View d. Sketch
3. What command is used to change the size of any selectable object?	a. Zoom b. Mirror c. Line d. Scale

PERFORM CADD FUNDAMENTALS

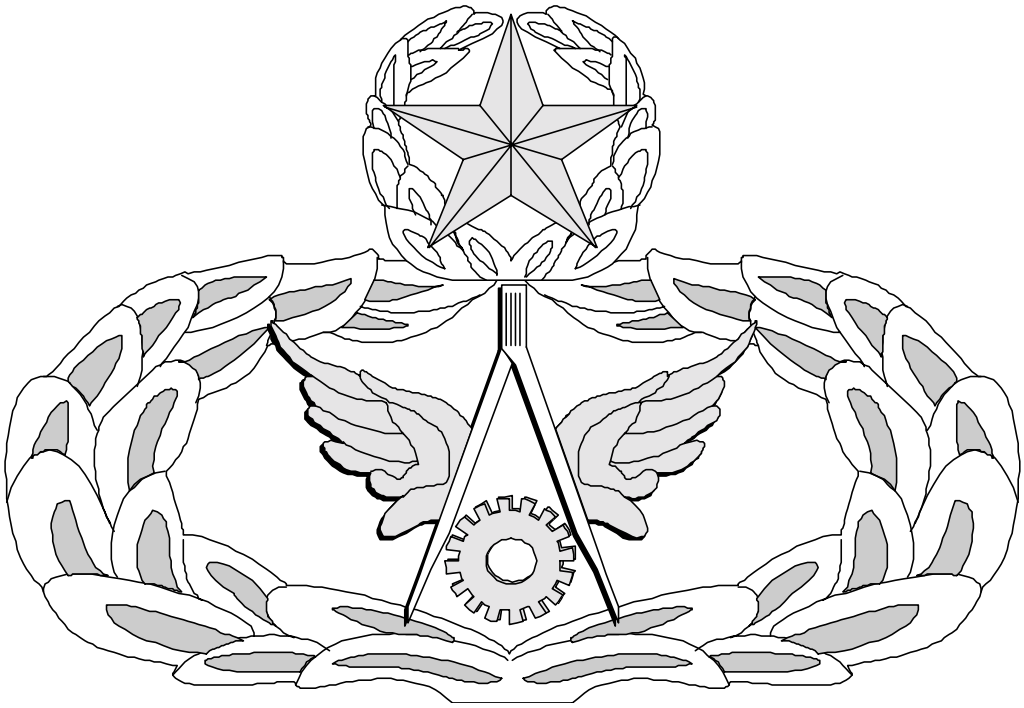
Performance Checklist		
Step	**Yes**	**No**
1. Was a new drawing started properly?		
2. Was the LINE command used properly?		
3. Was the CIRCLE command used properly?		
4. Was the COPY command used properly?		
5. Was the MIRROR command used properly?		
6. Was the ERASE command used properly?		
7. Was the UNDO command used properly?		
8. Was the trainee able to save the drawing?		
9. Was the floor plan successfully completed?		

FEEDBACK: Trainer should provide both positive and/or negative feedback to the trainee immediately after the task is performed. This will ensure the issue is still fresh in the minds of both the trainee and trainer.

Air Force Civil Engineer

QUALIFICATION TRAINING PACKAGE (QTP)

REVIEW ANSWER KEY



for

ENGINEERING

(3E5X1)

MODULE 15

COMPUTER AIDED DESIGN/DRAFTING (CADD)/ GEOGRAPHIC INFORMATION SYSTEM (GIS)

Notice. This AFQTP is *NOT* intended to replace the applicable technical references nor is it intended to replace hands-on training. It is to be used in conjunction with these for training purposes only.

PERFORM CADD FUNDAMENTALS

(3E5X1-15.1.1.)

Question	Answer
1. What are the five parts of the CADD program window?	b. Pull-down menu, toolbars, Drawing area, Command window, Status Bar
2. What toolbar contains the circle function?	b. Draw
3. What command is used to change the size of any selectable object?	d. Scale